Series Car **Brake Cooling**

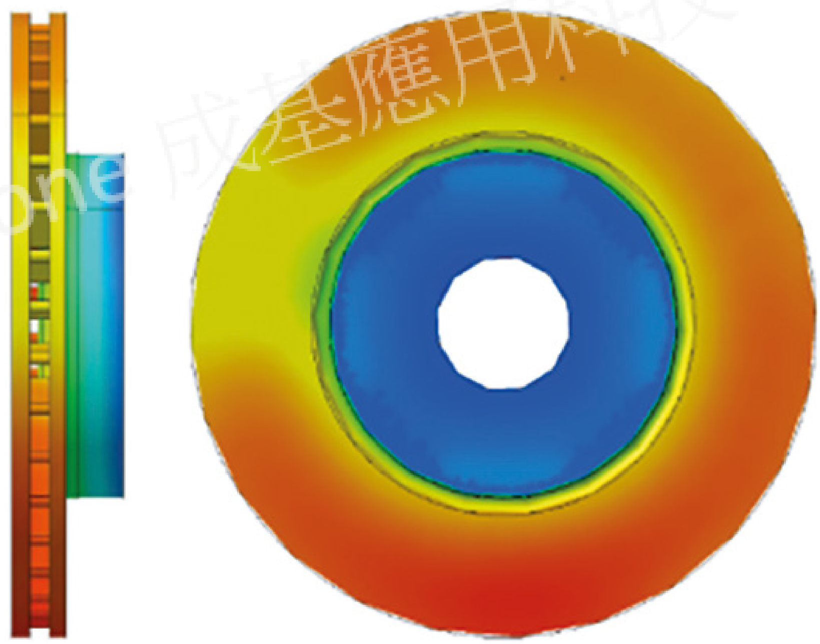
Test drives and simulating with FloEFD

By Marina Keskic, Master of Science (M. Sc.), Technische Hochschule Mittelhessen

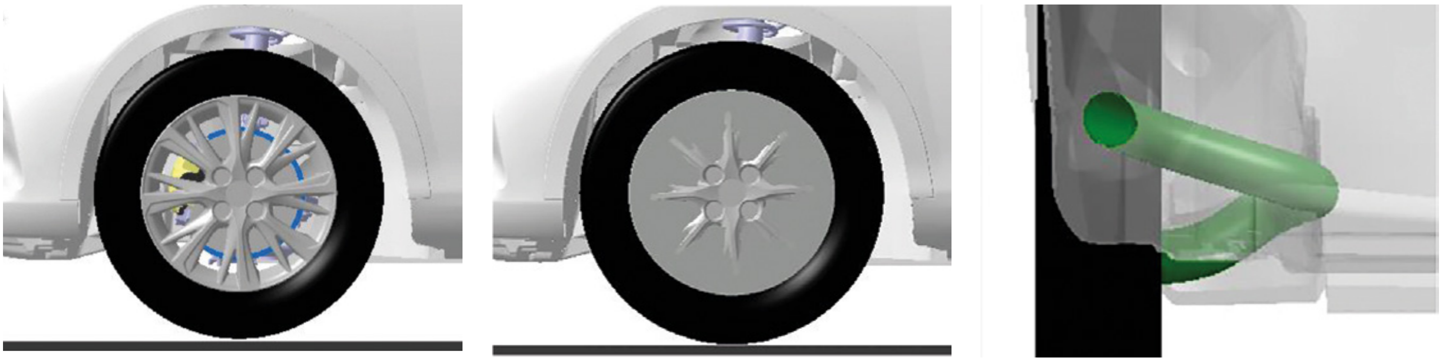
Passenger car brakes are components under high thermal loads. Good ventilation of the brake components and achieving adequate cooling behavior are therefore crucial. The thermal design of the braking system plays a significant role when considering the reliability of braking performance. The directed airflow into the wheelhouse can be in conflict with the aerodynamics of the entire vehicle.

During braking, kinetic energy is converted into heat by means of friction. The brake disc absorbs approximately 90% while the brake linings absorb approximately 10% of this heat. The brake components must be able to resist these high thermal loads in order to ensure consistent reliable performance of the brake. The heat absorbed mainly by the brake disc and brake linings is then dissipated by convection, heat conduction and radiation to the ambient air and the surrounding components such as brake caliper, rim, and wheel carrier.

Driving tests and CFD simulations with FloEFD™ embedded in CatiaV5 were carried out for six design variants, examining the influence on the cooling behavior of the brake disc. The intention of the study was to show the ability of CFD simulation to efficiently evaluate the brake cooling conditions for different configurations based on a phenomenological approach, under consideration that by the use of Computational Fluid Dynamics (CFD) simulations, the sources of possible functional impairments can be identified, evaluated and eliminated early in the vehicle and brake system development. Target for brake system development is that the simulation approach needs to be pragmatic, because of the early stage of car development. One goal was to determine which configurations have the greatest influence on the brake disc cooling capacity and how they affect the flow profile in the wheel housing.



The basic design, variant one, is shown in Figure 1, top left. Variant two was created to clarify the difference between an opened ventilation through the rim and a closed rim. For variant three, an air supply with a tube was installed at the closed rim. For this purpose, an aluminum flex hose with a diameter of 80 mm was installed through the wheel housing directly to the brake disc. For the fourth variant, the open rim was equipped with the hose for an additional air supply. In variant five, the splash shield behind the brake disc (see blue plate in variant one) was completely removed to



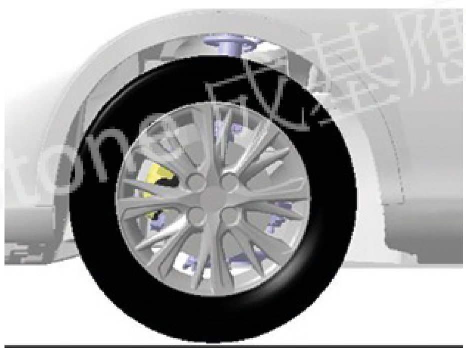
examine the cooling process without splash shield. The air supply through the hose was thereby maintained. In the sixth variant, the hose and the splash shield were removed. In comparison to the basic version, the influence of the splash shield on the brake disc cooling performance can be assessed by this configuration.

The test drives were carried out on the Continental test track. The brake disc was heated up to 400°C on the test track with dragging brakes while towing the car. After reaching the target temperature, the temperature level was held by dragging the brakes for another few minutes until the start of the cooling phase to obtain an evenly heated brake disc. The cooling process was started afterwards. For this purpose, the vehicle was driven at a constant speed of 50km/h. The cooling phase was carried out until the brake disc temperature reached 200°C. The temperature of the brake disc was measured at the friction ring. The ambient temperature was in the range of 21 to 26°C.

The evaluation of the driving tests showed that variant two produced the worst conditions for the brake disc cooling, which was expected. Variant five showed the best cooling behavior.

In the simulation, the brake cooling was analyzed for a constant speed of 50 km/h, at an ambient temperature of 23°C, with a uniform initial brake disc temperature of 400°C. The physical time period of the simulation was 200 seconds and 300 seconds respectively, depending on the variant.

The basic design model consists of the following parts: Car body, brake disc, caliper, splash shield, wheel bearing, axle, tire, rim, and road. Wheel, rim and wheel housing are particularly important as they have a high influence on the flow field and thus on the cooling behavior of the brake. In the simulation, a quarter of a section



Variant	Title
V01	Basic Design
V02	Closed Rim
V03	Closed Rim + Hose
V04	Hose
V05	Without Splash Shield + Hose
V06	Without Splash Shield

Figure 1. Six variants

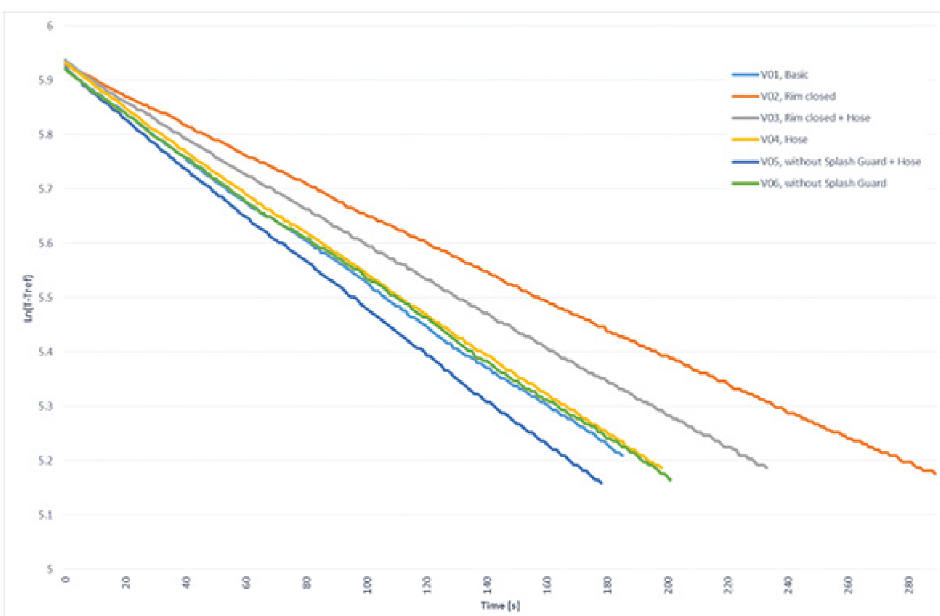


Figure 2. Results test drive

This article was originally published in Engineering Edge Vol. 7 Iss. 1

of the vehicle was examined, focusing on the brake disc and the wheel housing. The FloEFD project was defined as an external flow domain, so that the local flow field in the observation area can be well represented.

The “sliding” rotation approach was used for local regions. This multi-zone model of rotation allows for the simulation of cases where the flow field is transient and highly non-uniform around the rotating part. In the local rotating zone the calculation is performed in the local rotating reference frame, whose boundary slides along the static zone’s boundary and the information are exchanged at each time step. The rotating region is shown in Figure 5 and it was defined with an angular speed of 7.12 Hz, which corresponds to a vehicle speed of 50km/h, with a wheel radius of 0.31m. The car body, the road and the tire were defined as thermal insulators, because the thermal influence was considered as negligible in this study. The flow rate for the additional air supply was defined as 0.022m³/s at the beginning of the hose, resulting from a further simulation for an external flow with 50km/h.

The influence of the radiation was estimated in additional analytical calculations and simulations. These investigations demonstrated consistent results that heat conduction and convection have the biggest influence on the cooling performance for this simulation case with the existing temperatures. Therefore, the radiation was neglected for the simulation and none of the available FloEFD radiation models was applied.

The channels in the brake disc, the gap between wheel and car body and the openings in the rim are particularly important. For this reason, a local mesh refinement was defined for these areas, as well as for the area of the additional air supply in the variants three, four, and five.

The automatic simulation time was approximately eight hours, for the transient calculation of 300 physical seconds.

The comparison between the test results and the FloEFD simulations showed a good correlation, for the six investigated variants. Only variant five shows visible deviation, which can be explained by a geometrical deviation of the applied flex hose compared to the simulation model. The outlet of the hose was on a different axial position, caused by the fact that that there is no overlapping the rotational domain. In the test

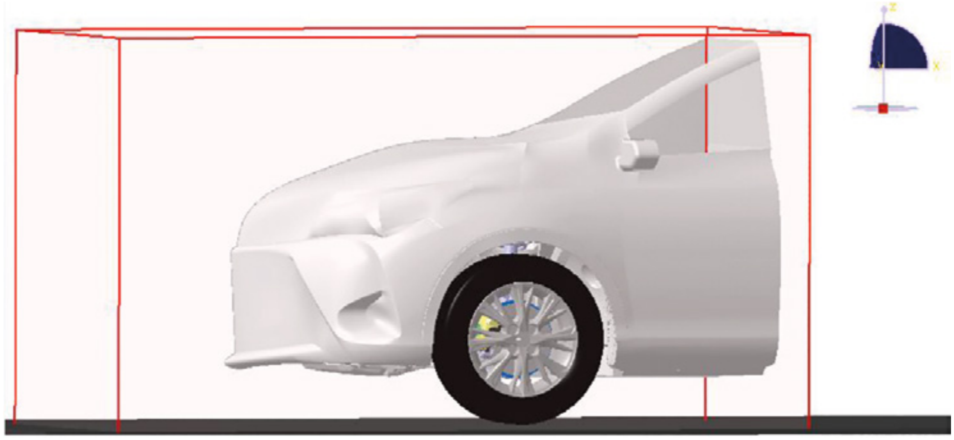


Figure 3. Computational domain for the simulation

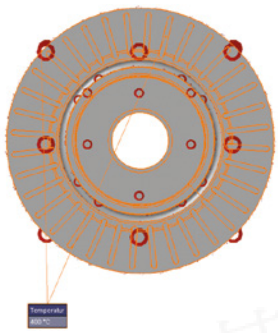


Figure 4. Initial temperature, brake disc

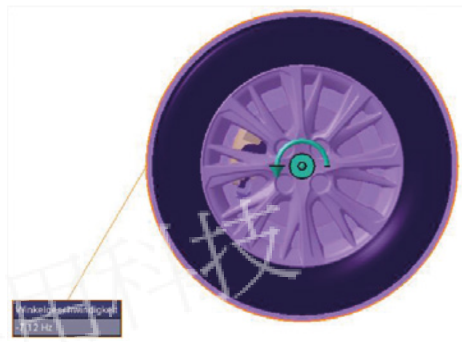


Figure 5. Rotation region

Figure 6 a+b. Additional air supply

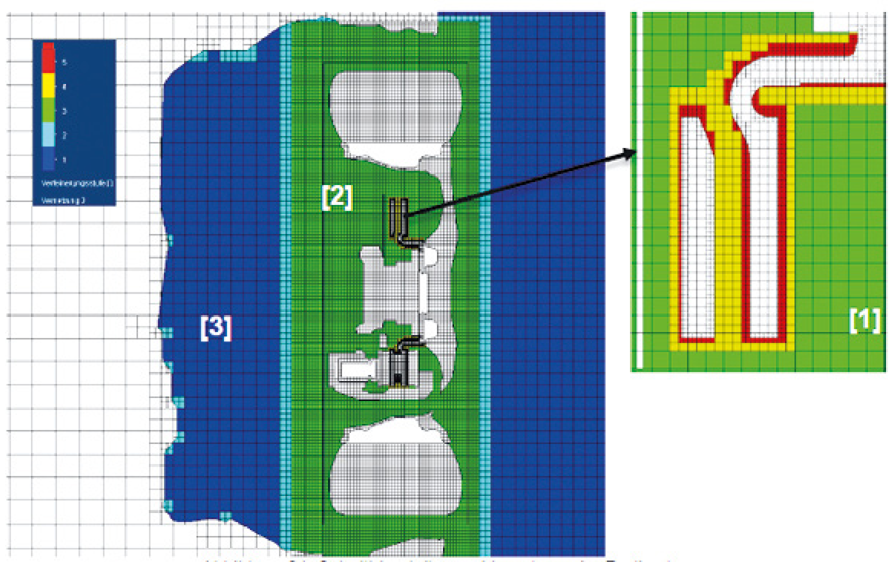


Figure 7. Cut plots of the mesh in the wheel house

configuration, the outlet was much closer to the disc than in simulation, which explains the better cooling performance.
A main advantage of a CAD embedded simulation is that, in addition to providing additional numerical results, it also enables graphical visualization of the interested parameters at any position directly in the CAD model, for velocity or pressure conditions for example.

The comparisons in Figure 12 show variant one (base), variant two (closed rim, worst case) and variant five (splash shield removed, but with hose for additional air supply), after 150 seconds each.

The figures show that in variant one the airflows from the inside outwards, i.e. the flow runs from the wheel house outwards into the rim and gets swirled in front of the brake disc. There are barely any areas without air movement. With variant two, the closed rim avoids air exchange and areas with very low air velocities (approximately 0km/h) can be identified. Variant five generates a very turbulent flow profile with partially higher velocities, but also shows areas with very low air velocities.

For variant one, the velocity in the cooling channels is higher than for variants two and five.

After 150 seconds, the brake disc in variant two had a maximum temperature of 278°C. For variant one, this was 241°C and variant five, with the best cooling conditions, at 233°C.

The closed rim clearly had the strongest influence on the cooling behavior of the brake disc. The test results showed that variant five, the brake disc had the fastest cooling down to the target temperature of 200°C. With this variant, the best conditions for cooling were given. The rim was open, an additional air supply through a hose was provided, and the splash shield was removed. Variant two, on the other hand, took the longest with 296 seconds. Due to the closed rim, this variant did not provide good conditions for the cooling behavior. A closed rim reduces the cooling of the brake disc by approximately 30% compared to the basic version. The removal of the splash shield and the installation of the hose does not improve the cooling behavior individually. The hose position must be selected in such a way that the additionally injected air flow supports or enhances the already existing flow in the wheel arch. In that case, variant four (with hose) would also achieve even better cooling characteristics.

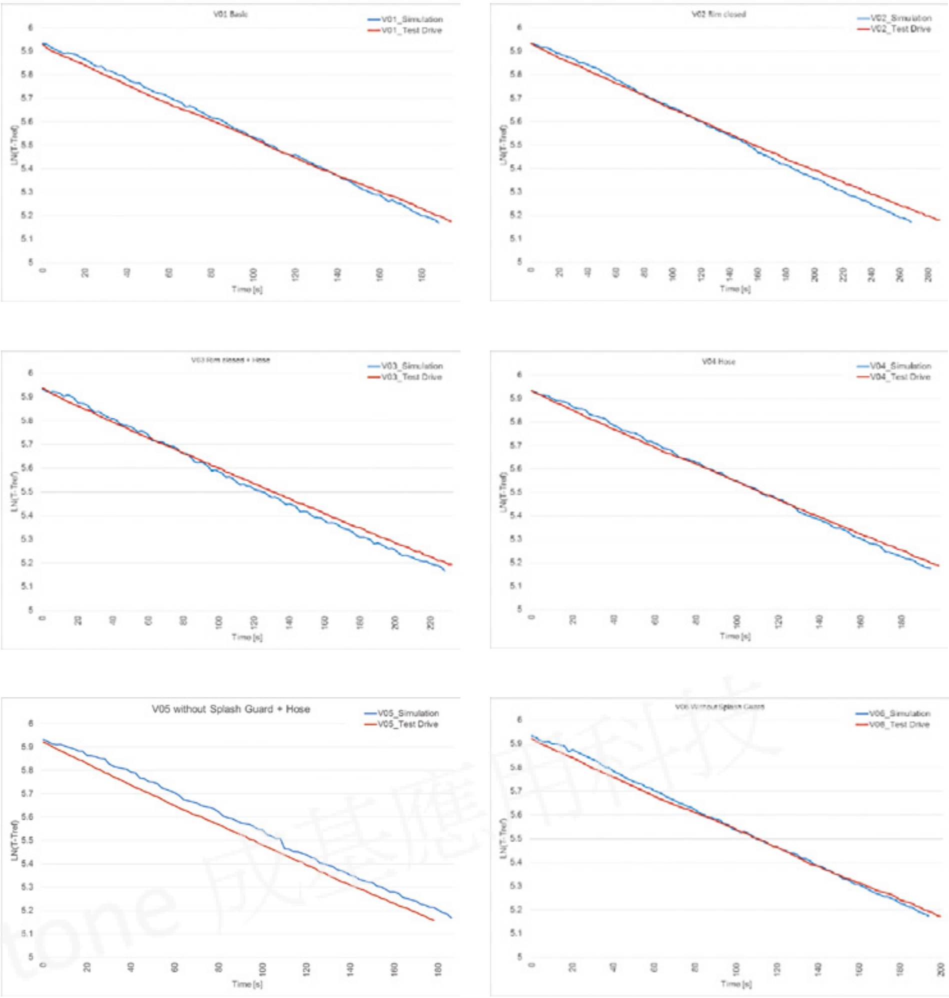


Figure 8. Comparison test drive and FloEFD Simulation

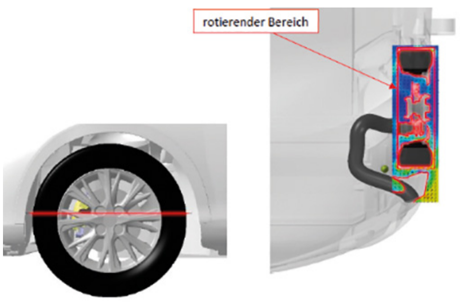


Figure 9. Section XY

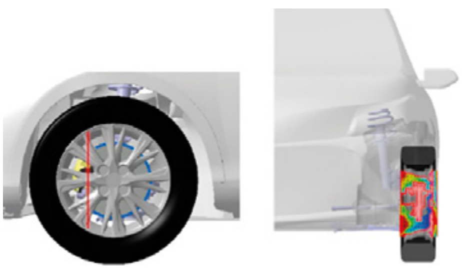


Figure 10. Section YZ

Figure 11. Section XZ

> "FloEFD provided a quick and easy introduction into flow analysis, due to the easy-to-use and engineering focused user interface. Good results are provided within short calculation times.
>
> **Marina Keskic, Master of Science (M. Sc.), Technische Hochschule Mittelhessen**

In variant five, where both (splash shield and hose) were carried out, the cooling performance of the brake disc improved by 10.2%.

The FloEFD simulations illustrated the flow behavior in the wheel house and led to a better understanding of the cooling behavior. The comparison with the test results showed a convincing agreement. The simulation also allows for the evaluation of further concepts and a directed airflow into the entire wheel house, taking into account the overall vehicle aerodynamics. Future optimizations may include, for example, the position of the air supply hose and the design of the splash shield in order to better support the natural airflow in the wheel house.

The investigation with FloEFD showed the ability to identify the drivers of the cooling behavior. To make a final rating the investigation on constant speed needs to be extended from steady state condition to unsteady condition on different speed levels.

References:

[1] Simulation der Luftströmung im Radkasten eines PKW zur Untersuchung des Abkühlverhaltens von Radbremsenkomponenten", Marina Keskic, Masterthesis Master of Science (M. Sc.), Technische Hochschule Mittelhessen, Friedberg 2017-10-30:

[2] https://www.continental-automotive.com/en-gl/Passenger-Cars/Chassis-Safety/Brakes/Hydraulic-Brakes/Foundation-Brakes

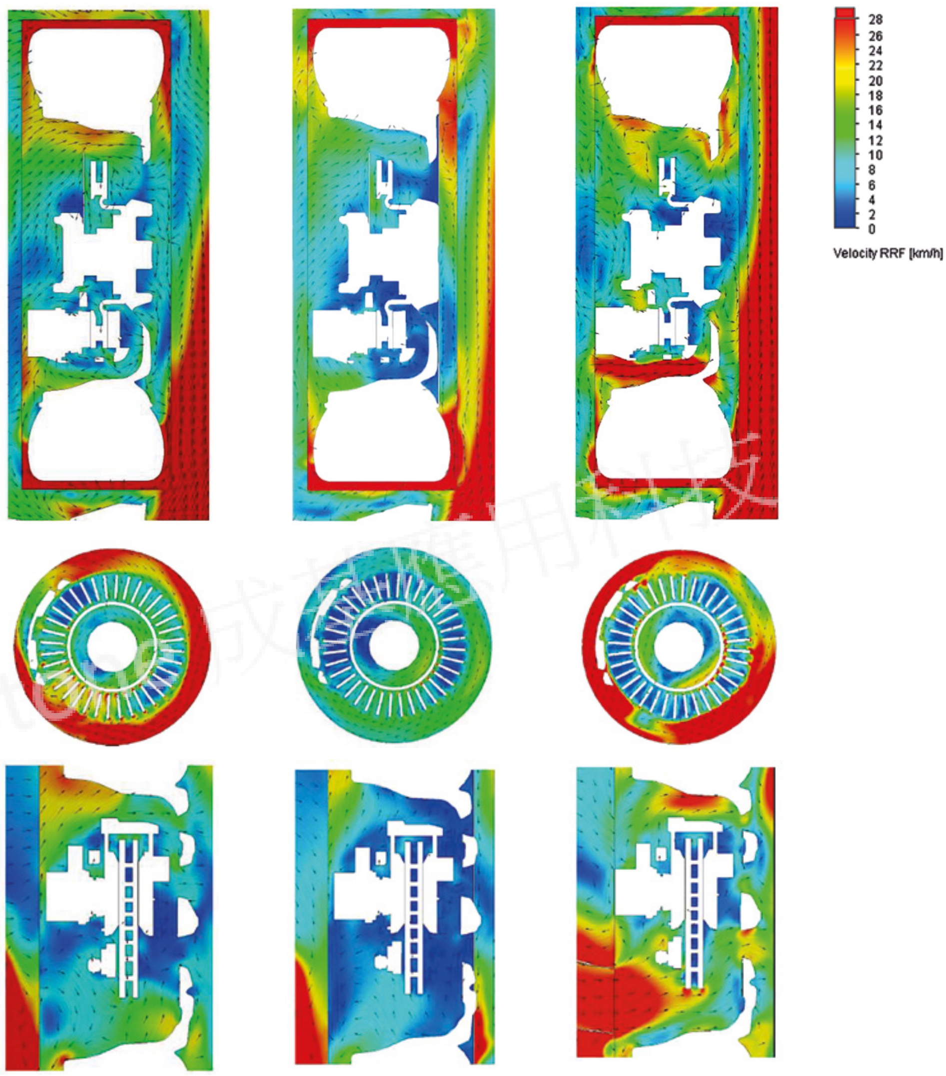


Figure 12. Variant 01, 02 und 05 (from left to right); section XY, YZ, ZX von (from top to bottom)

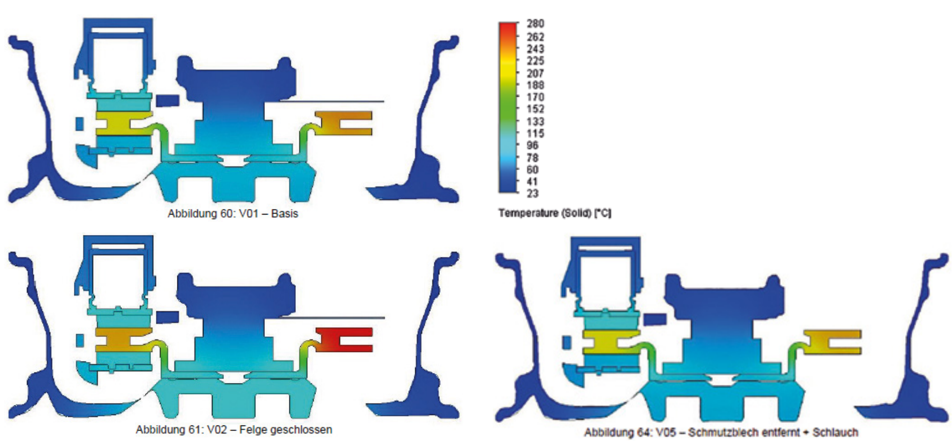


Figure 13. Temperature distribution for variants 01, 02 and 05 (from top to bottom)

Mentor®
A Siemens Business

This article originally appeared in Engineering Edge Vol. 7 Iss. 1

Download the latest issue:
www.mentor.com/products/mechanical/engineering-edge

ENGINEERING EDGE

Accelerate Innovation with CFD & Thermal Characterization

You might also be interested in...

Thermal Simulation of a SYN TRAC Powerpack

Edge Devices are Hot for IoT

Distributed Chilled Water Pumping AC System

Reliability Evaluation of Sintered Copper Die Bonding Materials

Virtual Reality and Modeling Tools

Ask the GSS Expert

Simulation of Hydraulic Servo Gimbal Control of a Space Launch Vehicle

How to… Convert CAD Geometry into a FloMASTER Sub-System

A Study of Aerothermal Loads

Development of a Turbocharger with FloEFD

Risk Prevention & Energy Saving in Ships

Interview: Takuya Shinoda, Denso Corporation

Three Thermal Simulation & Test Innovations for Electronics Equipment Design

Liquid Cooling Turned on its Side

Thermal Analysis of an ADAS Camera

Thermal Design Leading the Charge

Understanding Die Attach Thermal Performance

Rapid Beverage Cooling Analysis

Bitcoin Mining: A Thermal Perspective

Designing High Efficiency, Low-Cost Synchronous Reluctance Motors

From Measurements to Standardized Multi-Domain Compact Models of LEDs

A Cool Emulator

Achieving Precision in 1D Dynamic Models of Hydraulic Servo Valves

Using FloEFD as an Engineering Tool

Comparison between Experimental & Computational Results

Design of a High Speed Decoy UAV

Thermal Interface for Pluggable Optics Modules

Geek Hub: What do Engineers see in Coffee Grounds?